

Analysis of Turbocharger Performance for Jet Assisted Vertical Takeoff and Landing

Roopesh Kaimal¹, Jason Jacob², Deepu Mohan³

, Gokul G. Nair⁴, Libin P. Oommen⁵

¹²³⁴⁵Mechanical Department, Saintgits College Of Engineering, Kottayam, Kerala.

Abstract—This paper gives a brief analysis on the performance parameters of a Turbocharger, by fabricating a separate combustion chamber and mocking the working of a jet engine. Parameters such as variation of specific heat, dimensionless flow parameters, variation of turbulence, conductivity, thrust developed etc are studied using simulation of the model, and compared with the actual working of the prototype. It can be conveniently proposed from the experiment that turbocharger can be used effectively for developing vertical take-off assist.

Keyword—Turbocharger, VTOL, Performance, JATO, Take off, Jet Engine

I. INTRODUCTION

For the past 2 to 3 decades, enthusiasts and engineers around the world has been working all over the world for the invention of a car that can vertically take off into air. Such an invention would make possible a faster mode of transportation. The aviation industry has seen a hike in number of passengers over the last few years. It is expected that the number of passengers in airline will reach 3.6 billion by the year 2016(Source :IATA). Airline industries have also seen an increase in private flights around the world. All this show a future where flights are possible from the backyard of house. Many scientists has been researching on engines, aerodynamic effects, safety concerns etc of a possible VTOL aircrafts and vehicles.

In Advances in Combustion and Propulsion Applications[1], Mr.Candel and Mr.Durox has discussed on various combustion techniques like cryogenic combustion, Turbulent Combustion and the problems associated with the same. Using numerical simulations, they showed that by using turbulence as the prime driver of combustion, process of combustion can be improved to a greater extent.

Survey conducted by Jean P. Renaud for Air and Europe had presented various advances in technologies of VTOL aircraft using rotorcraft model. The main disadvantage of such an aircraft is with the propulsion system, capacity, size, sound and speed of the resulting aircraft.

In 2007, Researchers from university of Padova took the initiation to develop a small scale low cost turbojet engine. The results were published in Design, manufacturing and

operation of a small turbojet-engine for research purposes[3]. The design focused on low cost and high thrust. But the research didn't include the comparison of various performance parameters and the analysis of combustion taking place inside the turbojet engine.

In 2013, Analysis was conducted on small scale turbojet engine to determine the performance parameters[4]. Various parameters like mass flow rate, pressure, velocity etc were compared with experimental and numerical data and the results were compared in the paper.

Though there were many researches in turbojet engines, most of them concentrated on axial flow engines. But for use in a VTOL aircraft, centrifugal flow engines are more suitable. Centrifugal engines can direct flow in a 90degree angle enabling the jet to be directed to the ground for a vertical takeoff. Though many military aircrafts use engines where nozzle can be adjusted to redirect the flow, very few analyses have been done on the same.

In this paper, a turbojet engine model is fabricated using a turbocharger and numerical and experimental analysis are done on the model to determine the variation in performance parameters when used to take off a model vertically from the ground. The ultimate goal of this project is to develop a small scale economic jet engine that can develop high thrust.

II DESIGN

The specification of the prototype was selected such that an economic low cost engine can be developed for the experimental purpose. The following assumptions are made:

- A simple Brayton Joule Cycle was adopted for analysis of the whole purpose. This relieves the apparatus from any complication.
- Considering the turbine blade temperature safety limit, the maximum inlet temperature was fixed at 1000 K.
- Commercially available turbocharger was used for the portion of centrifugal compressor and turbine.

- Following set of parameters are assumed for the design purpose
 - Turbine Efficiency = 70%
 - Compressor Efficiency = 70%
 - Ambient condition : 290K temperature, 1 Bar pressure, 75% humidity
 - Cycle – Brayton joules Cycle
 - Combustion efficiency = 95%

III DESIGN PROCEDURE

Selection of Turbocharger

A KKK type turbocharger manufactured by TEL was selected for the purpose. The turbocharger had single stage compressor and turbine with an inlet double stage diffuser that reduced the velocity of the incoming air. Non waste gate turbocharger was selected for the purpose so that energy loss is reduced.

Selection Of Materials

The material for the connecting pipes, combustion chamber and flame tube were selected on the basis of material cost, density, availability etc. After comparing Mild Steel, Stainless Steel, Titanium and Aluminum, MS was chosen as the best option for the prototype.

Table 1. Comparison of properties of metals

Metal	Cost (Rs/Kg)	Melting Point	Density (g/cm ³)	Thermal Conductivity (W/m K)
Mild Steel	65-70	1450-1550	7.80-7.90	50.2
Stainless Steel	100-120	~1500	~ 8	
Titanium	300-3000	1650-1750	~4.5	15-20
Aluminium	100-110	~650	~2.7	~205.0

Gasoline was chosen as the best fuel alternative. This is because the compressor of the turbocharger was not heavy enough to provide the required compression ratio for diesel. Kerosene can lead to coke deposits inside the flow field when cooled down. These deposits can later give away from the walls to damage the turbine blades. Aviation Fuel was not available for commercial use in the market. LPG was harder to set up and the risk of backfiring was greater in LPG.

Design of Combustion Chamber

The combustion chamber was designed to withstand a safe stress limit that combined the thermal stresses induced during the combustion process, in addition to the pressure force developed inside the chamber during the explosion.

A Dual flow design was opted for the prototype. The outer chamber consisted of a smaller inner flame tube within which the combustion takes place. The flame tube was divided into three stages which include primary zone, secondary zone, and tertiary zone. Holes were drilled onto each section of the flame tube so that predetermined air flows through the holes. This is to ensure efficient and complete combustion of the fuel inside the combustion chamber.

The primary or the premixing zone consist of 20 equally distributed zones split into two rows such that about 21% of total intake air is permitted into flames tube, when the turbine is running at its rated speed of 60,000 rpm. This zone ensures a stoichiometric mixing of air and fuel in this zone.

The secondary zone consists of 6 equispaced holes around the periphery of the flame tube, so that 55% of air is permitted inside the flame tube to provide excess of air into the flame tube. This excess air makes the mixture lean ensuring a complete combustion and a higher thrust.

The tertiary or cooling zone had 5 holes equispaced along the periphery of the flame tube that ensured 34% of air to enter the flames tube. This zone acts as a quenching zone that reduces the flame temperature before it hit the turbine blades. It also helps in ensuring a stable flame by providing a longer path for the flame front to travel.



Fig. 1 Flame tube

The air entering the tertiary zone is swirled in between the combustion chamber and the flames tube. This serves for two purposes. Part of the circulated air carries away heat from the walls of combustion chamber. This heats up the air to a temperature higher than the ambient temperature, which will contribute to flame stability as it enters the flame tube through the tertiary zone.

A nozzle is provided at the end of the flame tube to convert the pressure head developed in the flames tube to Kinetic head inside the turbine. This reduced the temperature of the mixture, in addition to reducing the pressure of the mixture while flowing through the connecting pipes.

Placing Sparkplug and Fuel injector

To ensure a perfect combustion of the air-fuel mixture inside the flame tube, the sparkplug was positioned in a plane perpendicular to the fuel injector. The fuel injector was placed before the primary zone while the spark plug was placed inside the secondary zone. This ensures a perfect premixing before the actual combustion taking place.

To reduce the size of the engine, instead of an afterburner, a second spark plug was placed just at the end of the tertiary zone. This sparkplug ensures that any unburnt mixture is given enough energy for combustion. This also ensures that the flame front travelled until the extreme end of the flame tube.

NUMERICAL ANALYSIS

In order to verify the design, numerical analysis was conducted on the prototype model using CAD packages. A 3D model of the prototype was built in the computer and put through a series of tests.

Static pressure tests, thermal tests were conducted on various parts of the prototype to ensure that the product met the design

specifications. Fluid Flow analysis was done through the parts of Combustion Chamber to study the flow field parameters. In order to conduct Numerical analysis, following assumptions were made.

- The turbine and the compressor are running at its rated speed of 60,000 rpm.
- The flow through turbine and compressors are of not much important, as knowing the efficiency of the both, their parameters can be understood.
- Turbine extracts energy from the gas only to power the compressor to which it is coupled off. Rest of the energy is given out through the exhaust either as temperature, or kinetic or potential heads.
- Gasoline-Air mixture after combustion can be satisfactorily regarded as Ideal air with only 5% error in the assumption.
- As per the design, the Fuel-Air mixture is static in the direction of axis of the cylinder in the primary zone. This assumption is because the fuel and the air enters in a perpendicular direction and the air is made to surround the fuel spray from all around the periphery of flame tube.
- Complete combustion is taking place in the flames tube. There is no a deposit or unburnt gases escaping from the chamber.
- The turbine does no other work in the flow other than extracting some amount of energy from the combustion gases to power the compressor. Hence the flow through the turbocharger is of less importance.

Static analysis

In order to study the deformation of pipes under pressure and thermal stresses, the combustion chamber and the connecting pipes were tested for static pressure test and thermal stress.

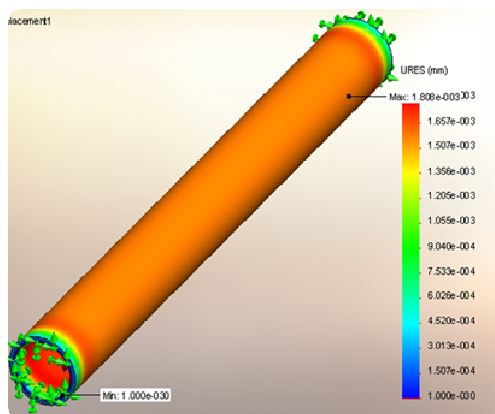


Fig. 2 Static Pressure test

Fig. 2 above shows the result of static pressure test. The deformation of the tubes was within the limits of the safe limit. The deformation was obtained to be around 1 micrometer.

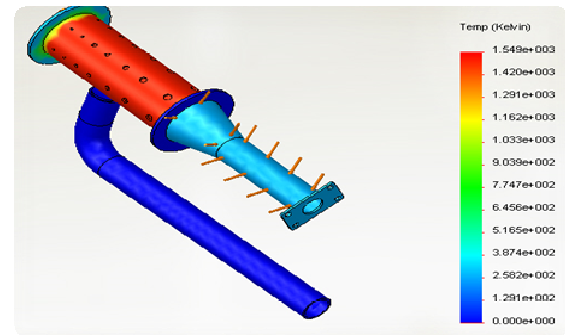


Fig. 3 Heat conduction in flames tube and adjacent pipes

Heat flow by conduction along the tubes under static condition was analyzed to determine the thermal stresses that might be induced in the pipes. Fig. 3 shows the result of temperature flow under static conditions. It can be concluded that not much heat is given to adjacent tubes due to the generation of heat inside the flame tube.

Fluid Flow Analysis

Air under ambient conditions was given into the compressor and the flame tube and the variation of flow parameters were analyzed inside the flame tube.

Following assumptions were made for the analysis

Ambient temperature = 288K

Ambient Pressure = 101.25 KPa

Humidity = 40%

- Inlet Conditions to the compressor are ambient conditions. There is no external work done on the air by any object other than the diffuser to the compressor. On a plane immediate to the walls of the diffuser, the conditions of the air are ambient.
- Gasoline is assumed to have 85% purity. The impurities in the gasoline though are assumed to have no effect in the flow field.

From the analysis, it was obtained that the velocity at the exit was close to sonic. A maximum temperature of 1500K was obtained in the flames tube while the temperature in the turbine blades was limited to below 1100 K. The temperature at the exhaust was limited to under 500 K, reducing the risk of thermal shocks to objects nearby. The two concentric cylinders reduced the thermal losses from the cylinder and maximum efficiency was obtained in combustion. Parameters like Thermal conductivity, specific heat and other dimensionless numbers were fluctuating across the nozzle in the flames tube. This might be because, at the left side of the flames tube there was a higher temperature and pressure head while towards the side of turbine, the conditions were lower temperature and a higher velocity. A maximum force of around 145 Kg was obtained from the engine from which it can be concluded that the engine can be used to demonstrate a volant of considerable weight. The intensity of turbulence was lower than expected. Hence turbulent energy loss was also lower.

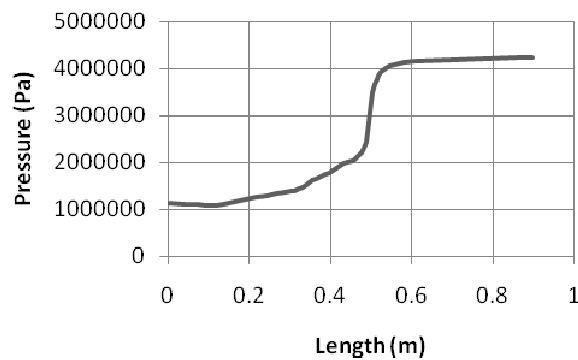


Fig. 4 Variation of Pressure along flame tube length

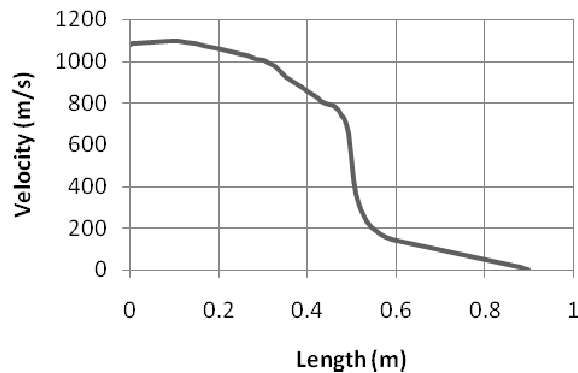


Fig. 5 Variation of Velocity along flame tube length

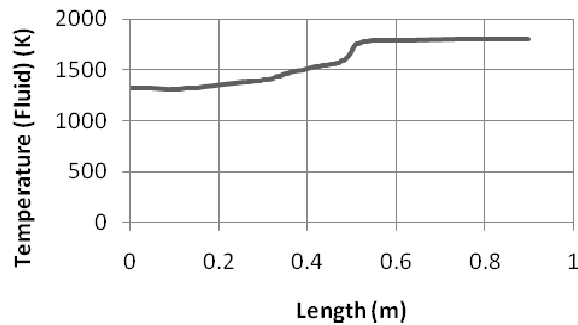


Fig. 6 Variation of Temperature along flame tube length

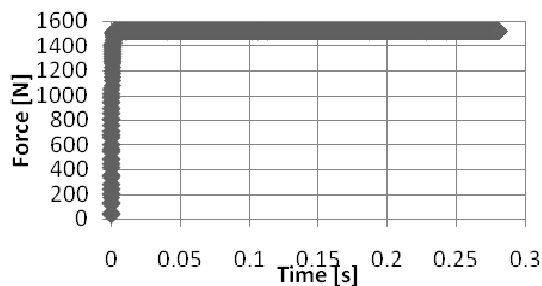


Fig. 7 Variation of Force with time

From the results of the numerical analysis, it can be concluded that the engine can generate suitable amount of thrust.

The pressure variation is shown in Fig. 4. The pressure has an extreme value of around 42 bar inside the combustion chamber. But it gradually reduces along the length until it steeply gets to 20 bar as it pass through the nozzle section. This reduces the temperature from around 1700 K to 1400K as can be inferred from Fig. 6. This temperature is further reduced when the effect of the turbine is also considered. Fig. 6 shows the variation of velocity along the length of flame tube. The velocity reaches around 1100K at the exit of the nozzle. But high temperature results in the mach number to be less than 1.

EXPERIMENT

In order to verify the results obtained in numerical analysis, experiment was conducted using the prototype built.

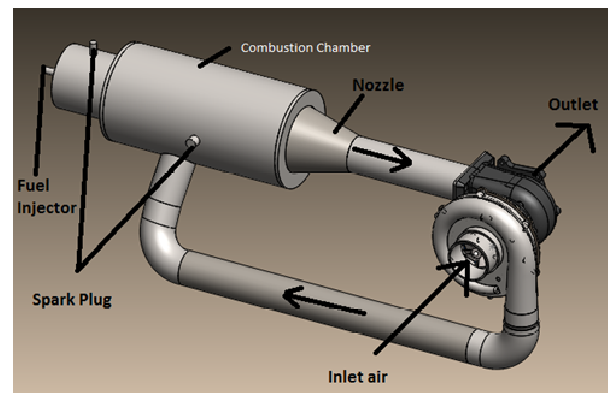


Fig. 8 Cad model of prototype

Fig. 7 shows the cad model of the prototype built using 3D modeling software. As marked, the air enters into the inlet of the compressor blades. The compressed air is then fed into the combustion chamber at high pressure. Fuel is sprayed into the combustion chamber perpendicular to the entry of air, to ensure proper mixing. The exhaust gases are fed to the turbine through a nozzle that serves to reduce the pressure and temperature of the gas hitting the turbine blades. The turbine after extracting energy to drive the compressor throws out the gases. These gases expand to atmospheric pressure resulting in thrust in the opposite direction as shown.



Fig.9 Final Fabricated model before testing

Based on the design and analysis results, a model was fabricated as shown in Fig. 9. The model was fabricated using

Mild Steel tubes. 3 HP 1450 rpm single phase electric motor was used to drive the pump of the fuel injector. Belt drive was used to drive the pump at its rated speed of 1750 rpm. K type thermocouple was used to measure the temperature at the exhaust as well as on the outside of the combustion chamber. As the temperature at the exhaust was small, standard pressure gauge was used to measure the exhaust pressure. The velocity was measured using a calibrated turbine placed at the exhaust of the turbocharger. The force was calculated using the thrust equation.

The experiment was conducted for two different mass flow rate of fuel at turbine rated speed of 60,000 rpm. The thrust obtained was 25Kg and 40 Kg respectively at a fuel flow rate of 2 LPM and 3 LPM. The temperature at exhaust was 350K and 500 K respectively and at the periphery of combustion chamber the temperature was 338K and 379 K respectively

CONCLUSION

In order to analyze the variation of parameters during the flow through a centrifugal jet engine and to determine the thrust developed from it, a jet engine was designed using an automotive turbocharger. Numerical and Experimental investigations are done on the turbocharger and the following conclusions can be obtained from the results obtained.

- By using limited funds and conventional methods, a high pressure of up to 45 bar and velocity of 1200m/s was obtained from the engine.
- The thrust developed by the engine during the experiment was 40Kg. But numerical analysis showed that the engine can develop up to 145Kg of thrust. Minor modifications such as including preheat for fuel, sealed combustion chamber, efficient turbine and compressor, higher thrust to weight ratio can be obtained.
- Proper design can reduce thermal and turbulence losses to a great extent. Hence higher efficiency can be obtained from the engine.
- The size of the engine can be reduced by using advanced technologies and non-conventional methods. This can yield a lower Thrust specific Fuel consumption.
- The fuel consumed by the engine was 120-180 Litres per minute when it developed 40 Kg of thrust. This can be improved satisfactorily.
- The exhaust temperature can be reduced to an extent that will not harm the objects nearby when the engine is powered.
- A low cost jet engine can be easily manufactured to yield a high thrust to weight ratio.

The understanding of subjects like flame stability is limited in the present scenario. This knowledge is to be widened in order for a more keen combustion in the engine. A deeper

understanding of various flow parameters are to be done

inorder to improve the engine efficiency and thrust produced.

FUTURE WORKS

In order to obtain more information on the flow properties, following researches are planned and suggested:

- Analysis of the engine for varying mass flow rates of fuel and air.
- Varying the fuel used for combustion. Using alternate fuels like Diesel, Propane, LPG, Kerosene, Aviation Fuel, Bio-Diesel and other alternates and studying their impact on the variation of properties of the engine.
- Including the analysis of flow in turbine and compressor.
- Including the effects of fuel impurities, thermal losses, incomplete combustion, turbulent energy losses, and vibrational losses.

A better understanding of these properties is necessary for further improvement in a low cost jet engine that can yield for a future mode of transport.

REFERENCES

- [1]. ChehhatAbdelmadjid, Si-Ameur Mohamed, BoumeddaneBoussad, CFD Analysis of the Volute Geometry Effect on the Turbulent Air Floq Through the Turbocharger Compressor, TerraGreen 13 International Conference 2013 – Advancements in Renewable Energy and Clean Environment, Elsevier, Energy Procedia 36 (2013) 746 – 755
- [2]. Ernesto Benini, Stefano Giacometti, Design, Manufacturing and operation of a small turbojet engine for research purposes, Elsevier, Applied Energy 84 (2007) 1102 – 1116
- [3]. M. Badami, P. Nuccio, A. Signoretto, Experimental and Numerical Analysis of a small scale Turbojet engine, Elsevier, Energy Conversion and Management 76 (2013) 225-233
- [4]. M. Deligant, P. Podeving, G. Descombes, Experimental identification of Turbocharger Mechanical Friction losses, Elsevier, Energy 39 (2012) 388 – 394
- [5]. David Scott Underwood, MS Thesis, Primary Zone Modelling of Gas Turbine Combustors, Massachusetts Institute of Technology, June 1999